Summary of NAMELIST input to OVERFLOW 2.

&GLOBAL (Global inputs for OVERFLOW)

NSTEPS	Number of (fine-grid) steps to advance solution. Use zero for input check. [0]
RESTRT	TRUE—Read restart flowfield from file *q.restart*. FALSE—Start from initial free-stream flowfield. [FALSE]
NSAVE	≥0—Save the overall solution to file *q.save* every how many steps. <0—Save the solution to file *q.step#* every how many steps. Note that files are saved as *q.step#* for dynamic or adaption cases regardless of the sign of NSAVE. [100]
SAVE_HIORDER	Controls whether Q^{n-1} data for 2^{nd}-order restarts is written to *q.save* and *q.step#* files: -1—Always include Q^{n-1}. 0—Never include Q^{n-1}. 1—Only include Q^{n-1} for final *q.save*. 2—Always include Q^{n-1} for *q.save*, never for *q.step#* if NSAVE<0. [2]
ISTART_QAVG	0—Do not save Q average/perturbation data. >0—Start saving Q average and *(rho,u,v,w,p)* perturbation data at step ISTART_QAVG. Write to file *q.avg* whenever *q.save* is written. Note that average/perturbation data starts fresh every run. [0]
NFLUSH	Flush output history files (*fomoco.out, resid.out, turb.out, species.out, rpmin.out, sixdof.out, animate.out*) every how many steps. [100]
NFOMO	Compute aerodynamic forces and moments every how many steps. [10]
NQT	Global turbulence model type declaration: 0—Algebraic or no turbulence model. 100—Baldwin-Barth (1-eq) model. 101—Spalart-Allmaras (1-eq) model with trip line specification. 102—Spalart-Allmaras (1-eq) model. 202—k-ω (2-eq) model (DDADI left-hand side). 203—SST (2-eq) model (DDADI left-hand side). 204—k-ω (2-eq) model (SSOR left-hand side). 205—SST (2-eq) model (SSOR left-hand side). [0]
NQC	Variable γ model type declaration (number of species): 0—Constant γ, 1-gas variable γ, or 2-gas variable γ with mixing based on stagnation enthalpy. ≥2—Multiple gas variable γ based on solution of NQC species continuity equations. [0]
MULTIG	Flag to enable/disable multigrid acceleration. [FALSE]
FMG	Flag to enable/disable grid sequencing. [FALSE]
FMGCYC(level#)	Number of steps to take on coarser grid levels during grid sequencing. Here index 1 is the coarsest level, 2 the next finer, etc. [300 for all coarse levels]
NGLVL	Number of multigrid and/or grid sequencing levels to use. [3]
TPHYS	Starting physical time (overrides value from *q.restart*). [not specified]
DTPHYS	Physical time-step (based on V_{ref}). [0]
NITNWT	Maximum number of Newton/dual subiterations per physical time-step (0 for no subiteration). [0]

FSONWT	1.0—First-order time-advance for Newton/dual subiteration. 2.0—Second-order time-advance for Newton/dual subiteration. Intermediate values allowed. [2.0]
ORDNWT	0—Do not limit number of Newton/dual subiterations. >0—Order of convergence to limit subiterations (use L_∞-norm(RHS)). <0— Order of convergence to limit subiterations (use L_∞-norm(ΔQ)). [0]
RF	Global coordinate system rotation speed (rad/time, based on V_{ref}). [0.]
RFAXIS	Global coordinate system rotation axis (1/2/3 corresponds to x/y/z-axis rotation). [3]
CDISC	TRUE—Expect to read a NAMELIST input file *overdisc.con*, containing CDISC inverse design control information. This file will be updated by OVERFLOW. FASLE—Do not read or write any CDISC information. [FALSE]
GRDWTS	TRUE—Use grid timing information in *grdwghts.restart* for MPI load-balancing, if available. (Equivalent to USEFLE in $GROUPS.) FALSE—Use normal load-balancing algorithm. [FALSE]
MAX_GRID_SIZE	0—Use automatic grid splitting algorithm for load-balancing. >0—Specified (weighted) size limit for split grids. <0—Do not split grids. [0] (Sets default MAXNB and MAXGRD in $GROUPS.)
NOBOMB	Inhibit writing *q.bomb* file if solution procedure fails. [FALSE]
CONSERVE_MEM	Conserve memory by recomputing metrics and regenerating coarse-level grids every iteration. [FALSE]
WALLDIST	0—Read precomputed wall distance from file *walldist.dat* (PLOT3D function file format). ±1—Simple computation of wall distance. ±2—Global wall distance calculation. If WALLDIST is negative, write wall distance file *walldist.save*. [1]
NWALL	Recompute global wall distance every NWALL steps. [0] *NWALL is currently ignored: global wall distance is computed on startup and after grid adaption only.
DEBUG	0—Normal run. 1—Write turbulence model debug file *q.turb* and quit. 2—Write timestep debug file *q.time* and quit. 3—Write residual debug file *q.resid* and quit. 4—Write grid adaption debug file *q.errest* and quit. [0]
STOPTIME_STEPX	PBS time limit reserve factor (multiplies max step time). [1.5]
STOPTIME_SEC	PBS time limit reserve time (sec). [0.]

&OMIGLB (Global inputs for OVERFLOW-D) (OVERFLOW-D only)

IRUN	0—Do a complete run. 1—Run only through off-body (brick) grid generation. 2—Run only through overset grid connectivity (DCF). [0]
I6DOF	0—Body motion is defined by user-defined USER6 routine. 1—Body motion is defined by inputs in $SIXINP. 2—Body motion is defined by GMP interface (files *Config.xml* and *Scenario.xml*). [0]

DYNMCS	Enable/disable body motion. [FALSE]
NADAPT	0—Do not regenerate off-body grids. >0—Regenerate off-body grids every NADAPT steps, based on geometry proximity and solution error estimation. <0—Regenerate off-body grids every –NADAPT steps, based on geometry proximity only. [0]
NREFINE	Number of off-body grid refinement levels allowed for solution adaption (NADAPT>0). [2]
ETYPE	Sensor function for grid adaption error estimate. [0] 0—Undivided second difference (squared) of flow variables Q(1-5). 1—Vorticity magnitude. 2—Undivided vorticity magnitude.
SIGERR	Solution error order for adaption. [2.0]
EREFINE	Solution error estimate level above which the grid will be refined. [$(1/8)^{SIGERR}$]
ECOARSEN	Solution error estimate level below which the grid will be coarsened. [$(1/8)^{SIGERR+2}$]
R_COEF	Coefficient of restitution for collisions. [1.0]
LFRINGE	|LFRINGE| is the number of fringe points for near-body grids and hole boundaries. If LFRINGE<0, do not revert double- and higher-fringe orphan points to field points. [Determined from numerical scheme (all grids).]
IBXMIN,IBXMAX	Boundary condition type for X_{min}, X_{max} far-field boundaries. [47]
IBYMIN,IBYMAX	Boundary condition type for Y_{min}, Y_{max} far-field boundaries. [47]
IBZMIN,IBZMAX	Boundary condition type for Z_{min}, Z_{max} far-field boundaries. [47]
LAMINAR_OB	Force laminar flow in off-body grids. (Applicable to all 1- and 2-eq turbulence models except NQT=101.) [FALSE]

&GBRICK (Off-body grid generation inputs) (OVERFLOW-D only)

OBGRIDS	Allow or inhibit off-body grids. [TRUE]
MAX_BRICK_SIZE	>0—Maximum off-body grid size. ≤0—No limit on off-body grid size. [IGSIZE/2]
DS	Spacing of level-1 (finest) off-body grids. [must be specified]
DFAR	Distance to far-field boundaries. [5.]
XNCEN,YNCEN ZNCEN	Center of off-body grid system. Must be specified for repeatable off-body grid generation with body motion. [center of near-body grids]
CHRLEN	Characteristic body length for off-body grid generation. [Currently not used.] [1.]
I_XMIN,I_XMAX	0—X_{min}, X_{max} far-field boundary will be determined by DFAR. 1—X_{min}, X_{max} boundary will be specified by P_XMIN, P_XMAX, resp. (Only one may be specified in the x-direction.) [0]
I_YMIN,I_YMAX	0—Y_{min}, Y_{max} far-field boundary will be determined by DFAR. 1—Y_{min}, Y_{max} boundary will be specified by P_YMIN, P_YMAX, resp. (Only one may be specified in the y-direction.) [0]
I_ZMIN,I_ZMAX	0—Z_{min}, Z_{max} far-field boundary will be determined by DFAR. 1—Z_{min}, Z_{max} boundary will be specified by P_ZMIN, P_ZMAX, resp. (Only one may be specified in the z-direction.) [0]

P_XMIN,P_XMAX	Physical location for X_{min}, X_{max} off-body grid boundary, if corresponding I_XMIN, I_XMAX≠0. [0.]
P_YMIN,P_YMAX	Physical location for Y_{min}, Y_{max} off-body grid boundary, if corresponding I_YMIN, I_YMAX≠0. [0.]
P_ZMIN,P_ZMAX	Physical location for Z_{min}, Z_{max} off-body grid boundary, if corresponding I_ZMIN, I_ZMAX≠0. [0.]
MINBUF	Minimum buffer width of points at each level. [4]
OFRINGE	Number of fringe points for off-body grids. [Determined from off-body numerical scheme or from *brkset.restart* file.]

&BRKINP (User-specified proximity regions) (OVERFLOW-D only)

NBRICK	Number of user-specified proximity regions. If NBRICK<0, user must specify ALL proximity regions (i.e., geometry will not be used). [0]
XBRKMIN(brick#), XBRKMAX(brick#)	X-range of user-specified proximity region.
YBRKMIN(brick#), YBRKMAX(brick#)	Y-range of user-specified proximity region.
ZBRKMIN(brick#), ZBRKMAX(brick#)	Z-range of user-specified proximity region.
BRKLVL(brick#)	Grid level for proximity region, in the range [1,-NREFINE]. [1]
IBDYTAG(brick#)	0—Proximity region will have no body transformations. >0—Proximity region will be linked to this body ID for dynamic motion. [1]
DELTAS(brick#)	Distance to expand proximity region (in all directions). [0.]
XREFMIN(region#), XREFMAX(region#)	X-range of error adaption limit region.
YREFMIN(region#), YREFMAX(region#)	Y-range of error adaption limit region.
ZREFMIN(region#), ZREFMAX(region#)	Z-range of error adaption limit region.
REFLVL(region#)	Minimum grid level for limit region, in the range [2,-NREFINE]. 0—Special value to freeze grid refinement in the limit region. [2]
IBDYREF(region#)	0— Limit region will have no body transformations. >0—Limit region will be linked to this body ID for dynamic motion. [0]
REFINOUT(region#)	"INSIDE"—Grid level will be limited inside this region. "OUTSIDE"—Grid level will be limited outside this region. ["OUTSIDE"]

&GROUPS (Load balance input) (OVERFLOW-D only)

USEFLE	TRUE—Use grid timing information in *grdwghts.restart* for MPI load-balancing, if available. (Equivalent to GRDWTS in $GLOBAL.) FALSE—Use normal load-balancing algorithm. [FALSE]
MAXNB	0—Use automatic splitting algorithm for near-body grid load-balancing. >0—Specified (weighted) size limit for split grids. <0—Do not split grids. [0] (Can be set by MAX_GRID_SIZE in $GLOBAL.)

MAXGRD	0—Use automatic splitting algorithm for off-body grid load-balancing. >0—Specified (weighted) size limit for split grids. <0—Do not split grids. [0] (Can be set by MAX_GRID_SIZE in $GLOBAL.)
WGHTNB	Weight-factor for near-body grids vs. off-body grids in normal load-balancing algorithm. [1.]
IGSIZE	Maximum group size during off-body grid adaption. [10,000,000]

&DCFGLB (DCF input) (OVERFLOW-D only)

DQUAL	Acceptable “quality” of donor interpolation stencils. [1.]
MORFAN	1/0—Enable/disable viscous stencil repair. [0]
NORFAN	Number of points above viscous wall subject to viscous stencil repair. [5]

&XRINFO (X-ray input, repeat per X-ray cutter) (OVERFLOW-D only)

IDXRAY	X-ray to be used for this cutter. (Note that X-rays may be used in multiple cutters.) [must specify]
IGXLIST	Specify a list of grids to be cut by this cutter (a grid number of -1 refers to all off-body grids). [none]
IGXBEG,IGXEND	Or specify beginning and ending grids to be cut by this cutter. [none]
XDELTA	Hole will extend XDELTA from the X-rayed surface. [0.]

&SPLITM (Write SPLITMX/SPLITMQ grid and/or Q files. Use multiple namelists as needed.)

XFILE	Write grid file to *<XFILE>.step#*. If XFILE is blank, don’t write grid file. [“ ”]
QFILE	Write Q file to *<QFILE>.step#*. If QFILE is blank, don’t write Q file. [“ ”]
QAVGFILE	Write Q-average file to *<QAVGFILE>.step#*. If QAVGFILE is blank, don’t write Q-average file. [“ ”]
NSTART	Start writing files at step NSTART (use –1 for last). [1]
NSTOP	Stop writing at step NSTOP (use –1 for last). [-1]
NSAVE	Save files every NSAVE steps. [1]
IPRECIS	0—Write files in default-precision (same as *grid.in*). 1—Write files in single-precision. 2—Write files in double-precision. [0]
IG(subset#)	Grid numbers for subsets. Near-body or off-body grid subsets can be specified using J,K,L indices; off-body grid cuts can be specified using CUT and VALUE.
JS,JE,JI(subset#)	J-index start/end/increment for subsets. [Increment default is 1]
KS,KE,KI(subset#)	K-index start/end/increment for subsets. [Increment default is 1]
LS,LE,LI(subset#)	L-index start/end/increment for subsets. [Increment default is 1]
CUT(subset#)	Off-body grid cut type (“x”, “y”, or “z”).
VALUE(subset#)	Coordinate value for off-body grid cut.

&ADSNML (Acoustic Data Surface parameters)

PGRID	List of 2D grids to save. Grids will be written in the same order as specified. [none]

PSAVE	>0—Save ADS files *ads_files/ads.save.[xq]* every how many steps. <0—Save ADS files *ads_files/ads#####.[xq]* every how many steps. =0—No ADS files will be written. [0]
PS,PE	Specify starting and ending step numbers for ADS file saves. PS=0 means start at the beginning of the run, PE=0 means save files until the end of the run. [0,0]
IPRECIS	0—Write ADS files in default-precision (same as *grid.in*). 1—Write ADS files in single-precision. 2—Write ADS files in double-precision. [0]

&CNDTNS (Rotor global parameters; ***rotorcraft coupling only***)

IFORMAT	1—Use standard rotorcraft interface files. 2—Use FSI format rotorcraft interface files. [1]
IMANVR	0—No vehicle motion. 2—Vehicle motion, approximated by time metrics for all grids. [0]
CFDORIGIN(3)	Origin of CFD coordinate system, in vehicle frame. [(0,0,0)]
CFDXAXIS(3)	X-axis of CFD coordinate system, in vehicle frame. [(1,0,0)]
CFDYAXIS(3)	Y-axis of CFD coordinate system, in vehicle frame. [(0,1,0)]

&EROTOR (Rotor parameters, one namelist per rotor; ***rotorcraft coupling only***)

ICOUPLING	<0—No aeroelastic coupling *and* no rotor force reporting. 0—No aeroelastic coupling. 1—Aeroelastic coupling using Euler angles. 2—Aeroelastic coupling using Euler parameters. [1]
DYMORE	Enable DYMORE structures code interface (not used). [FALSE]
TIPEXTRAP	Extrapolate aeroelastic deformations to the tip, if not defined. [TRUE]
ROOTEXTRAP	Extrapolate aeroelastic deformations to the root, if not defined. [TRUE]
IBS	Body ID (or component ID) for first blade. [1]
IBE	Body ID for last blade (blades are assumed to be consecutive). [1]
IGS	Main grid (used for force and moment calculations) for first blade. [1]
IGINC	Increment between main blade grids (assumed constant for all blades). [3]
NPSI	Number of azimuth stations for saving sectional blade forces to *<FORCEFILE>.onerev.txt*. [72]
IVISC	1/0—Do/don't include viscous forces. [1]
ITINC	Save rotor forces every ITINC steps. Sectional and total forces are written to *<FORCEFILE>.forces.txt*; total forces are written to *<FORCEFILE>.hist.txt*. [100]
IEXCH	>0—Exchange data with comprehensive code every IEXCH steps. ≤0—No data exchange. [-1]
NMAP	Inverse-maps updated every NMAP steps. [25]
NXRAY	X-rays updated every NXRAY steps. [NMAP]
RTIP	Rotor tip radius (in grid units). [1.]
MTIP	Tip Mach number (for blade force non-dimensionalization). [REFMACH]
CREF	>0—Reference chord (for blade force non-dimensionalization). ≤0—Use local blade section chord. [-1.]

RRCMREF(station#)	Radial stations for blade moment reference line, normalized by RTIP. (If RRCMREF<0, values are assumed to be in grid units.) [(0,1)]
XCCMREF(station#)	Chordwise offset from the leading edge of the blade moment reference line, normalized by the local chord. [(0.25,0.25)]
YCCMREF(station#)	Normal offset from the leading edge of the blade moment reference line, normalized by the local chord. [(0,0)]
SCALEMOTION(1:6)	Scale factor for each component of blade deformation (dx,dy,dz,roll,pitch,yaw). Used for clockwise-rotating rotors (set all components to –1). [(1,1,1,1,1,1)]
BASENAME	Base name for all rotor files. [*rotor_n*]
MOTIONFILE	Motion file for reading blade deformations. [*<BASENAME>.motion.txt*]
ONEREVFILE	Output file for blade sectional force map. [*<BASENAME>.onerev.txt*]
FORCEFILE	Output file for blade sectional forces. [*<BASENAME>.forces.txt*]
HISTFILE	Rotor total force history. [*<BASENAME>.hist.txt*]
QCFILE	Quarter-chord diagnostic file. [*<BASENAME>.qc.txt*]

&FLOINP (Flow parameters)

FSMACH	Freestream Mach number (M_∞). [0.0]
REFMACH	Reference Mach number (M_{ref}). [FSMACH]
ALPHA	Angle-of-attack (α), deg. [0.0]
BETA	Sideslip angle (β), deg. [0.0]
REY	Reynolds number (Re) (based on V_{ref} and grid length unit). [0.0]
TINF	Freestream static temperature (T_∞), deg. Rankine. [518.7]
GAMINF	Freestream ratio of specific heats (γ_∞). [1.4]
PR	Prandtl number (Pr). [0.72]
PRT	Turbulent Prandtl number (Pr_t). [0.9]
RETINF	Freestream turbulence level $(\mu_t/\mu_l)_\infty$ for 1- or 2-eq turbulence models. [0.1]
XKINF	Freestream turbulent kinetic energy (k_∞/V_{ref}^2) for 2-eq turbulence models. [10^{-6}]
TARGCL	Enable the target C_L-driver option. [FALSE]
CLTARG	Value of C_L the code will try to match. [0.0]
CLALPH	Fixed value of $dC_L/d\alpha$ used to update ALPHA. [0.1]
NTARG	Number of steps between ALPHA corrections, with the following exceptions: corrections are not done during grid sequencing, and corrections are not done on the first or last fine-grid steps. [10]
CTP	Rotor thrust coefficient (for BC type 37). [0.0]
ASPCTR	Rotor radius (for BC type 37). [1.0]
FROUDE	Froude number (gravity term) (Fr) (based on V_{ref} and grid length unit). [0.0]
GVEC(1:3)	Unit up-vector for FROUDE gravity term (Note that this vector is taken verbatim—it is *not* modified internally by the angle-of-attack, since other orientation angles (such as bank angle) are not known.) [0,0,1]

&VARGAM (Variable γ input)

IGAM	Options for specifying calculation of γ when *not* solving species continuity equations (i.e., NQC<2): 0—Use a constant γ value of GAMINF. 1—Single gas with temperature variation of γ computed using ALT0-4, AUT0-4. 2—Two gases with temperature variation of γ computed using ALT0-4, AUT0-4; all gas 1 below HT1, all gas 2 above HT2, linear mix in between. [0]
HT1	Total enthalpy ratio $h_0/h_{0\infty}$ below which the mixture is all gas 1. [10.]
HT2	Total enthalpy ratio $h_0/h_{0\infty}$ above which the mixture is all gas 2. [10.]
SCINF(gas#)	Freestream species mass fraction $c_{i\infty}$. [1 for gas 1, 0 for all others]
SMW(gas#)	Species molecular weight MW_i, or normalized molecular weight MW_i/MW_∞ (if preferred). [1.0]
ALT0(gas#)	Lower temperature range polynomial coefficient a_0 (540°R<T<1800°R). [γ_∞ /(γ_∞ -1)]
ALT1(gas#)	Lower temp range polynomial coefficient a_1 (540°R<T<1800°R). [0.0]
ALT2(gas#)	Lower temp range polynomial coefficient a_2 (540°R<T<1800°R). [0.0]
ALT3(gas#)	Lower temp range polynomial coefficient a_3 (540°R<T<1800°R). [0.0]
ALT4(gas#)	Lower temp range polynomial coefficient a_4 (540°R<T<1800°R). [0.0]
AUT0(gas#)	Upper temperature range polynomial coefficient a_0 (1800°R<T<9000°R). [ALT0(gas#)]
AUT1(gas#)	Upper temp range polynomial coefficient a_1 (1800°R<T<9000°R). [0.0]
AUT2(gas#)	Upper temp range polynomial coefficient a_2 (1800°R<T<9000°R). [0.0]
AUT3(gas#)	Upper temp range polynomial coefficient a_3 (1800°R<T<9000°R). [0.0]
AUT4(gas#)	Upper temp range polynomial coefficient a_4 (1800°R<T<9000°R). [0.0]
SIGL(gas#)	Laminar diffusion coefficient σ_l. [1.0]
SIGT(gas#)	Turbulent diffusion coefficient σ_t. [1.0]

The following NAMELISTs are repeated per grid.

&GRDNAM (Grid name)

NAME	Grid name (not used internally). [blank]

&NITERS (Subiterations per grid)

ITER	Number of flow solver iterations per step. (Each flow solver iteration performs ITERT turbulence model iterations and ITERC species equation iterations.) [1]

&METPRM (Numerical method selection)

IRHS	0—Central difference Euler terms. 2—Yee symmetric TVD scheme. 3—Liou AUSM+ flux split scheme. 4—Roe upwind scheme. 5—HLLC upwind scheme. 6—HLLE++ upwind scheme. [0]

ILHS	0—ARC3D Beam-Warming block tridiagonal scheme. 1—F3D Steger-Warming 2-factor scheme. 2—ARC3D diagonalized Beam-Warming scalar pentadiagonal scheme. 3—LU-SGS algorithm. 4—D3ADI algorithm with Huang subiteration. 5—ARC3D Beam-Warming with Steger-Warming flux split jacobians. 6—SSOR algorithm (with subiteration), 32-bit arithmetic. 7—SSOR (Jacobi in L instead of J), 32-bit arithmetic. [2]
ILHSIT	Number of subiterations for D3ADI or SSOR. [10 for ILHS=6,7; 3 for ILHS=4]
IDISS	2—ARC3D dissipation scheme (2^{nd}-, 4^{th}-order dissipation on RHS and LHS). 3—TLNS3D dissipation scheme (same as IDISS=2, but smooth ρh_0 instead of ρe_0). 4—Matrix dissipation scheme (see additional parameters VEPSL, VEPSN, ROEAVG in $SMOINP). [3]
ILIMIT	Limiter for upwind Euler terms (IRHS=3-6): 1—Koren limiter. 2—Minmod limiter. 3—van Albada limiter. 4—WENO5M scheme (FSO>3 only). See DELTA for further control. [1]
BIMIN	1.0—Disable low-Mach preconditioning. -1.0—Enable low-Mach preconditioning; reset BIMIN to 3xM_{ref}^2. <1.0—Enable low-Mach preconditioning with specified BIMIN. [1.0]
SSOR_RELAX	Relaxation factor for SSOR schemes (flow eqns, turb models, species eqns). [0.9]
Q_LIMIT	TRUE—Limit Q update procedure to try to keep density and energy from going negative. FALSE—Use simple Q update procedure (from OVERFLOW 1.8). [TRUE]
MULTIG	Local flag to enable/disable multigrid acceleration for this grid. [Default is MULTIG value from $GLOBAL]
SMOOP	Smoothing coefficient for prolongation of coarse-grid solution onto next-finer level during grid sequencing. [0.0]
SMOOC	Smoothing coefficient for multigrid correction before interpolation onto next-finer level. [0.0]
SMOOR	Smoothing coefficient for multigrid residual before restricting to the next-coarser level. [0.0]
CORSVI	Enable/disable computation of viscous terms on coarse grid levels. [TRUE]
RECMUT	Recompute μ_t on finest level during multigrid. [FALSE]

&TIMACU (Time accuracy)

ITIME	Time-step scaling flag: 0—Constant time-step, no scaling (used for simple time-stepping or Newton subiteration). 1—Local time-step scaling (with 0.005 dimensional fudge-factor). 2—Local time-step scaling (with no fudge-factor). 3—Constant CFL number (based on CFLMAX value). This uses the sum of the (max) eigenvalue in each coordinate direction to determine the local CFL number. All other uses of CFLMIN and CFLMAX use the maximum eigenvalue to determine the CFL number. 4—Same as ITIME=3, but adjust timestep scaling based on local cell Reynolds number. (For ITIME=3-4 (and BIMIN=1), a robust timestep modification scheme for low-density regions uses CFLMIN/CFLMAX as a minimum scaling factor. Default factor is 0.01.) [1]
DT	Time-step factor. [0.5]
CFLMIN	Minimum CFL number. [0.0]
CFLMAX	Maximum CFL number. [0.0]
TFOSO	Order of time-accuracy, when using simple time-stepping (NITNWT=0): 1.0—1^{st}-order time-accuracy (Euler implicit scheme). 2.0—2^{nd}-order time-accuracy (trapezoidal scheme). Other values allowed; 0.5, 1.9 are OK. [1.0]

&SMOACU (Smoothing parameters)

ISPEC	Dissipation scaling flag; single value to specify ISPECJ,ISPECK,ISPECL: -1—Sum spectral radii in J, K, and L. 1—Constant coefficient dissipation. 2—Spectral radius in J, K, or L. 3—Weighted average of J, K, L spectral radii (TLNS3D-type). [2]
SMOO	0.0—Spectral radius is computed normally, as $\lvert U\rvert+kc$. 1.0—Sound speed c is replaced by $\lVert V\rVert/M_{ref}$, reducing smoothing in low-speed regions. Intermediate values are allowed. [1.0]
DIS2	2^{nd}-order smoothing coefficient. [2.0]
DIS4	4^{th}-order smoothing coefficient. [0.04]
FSO	Order of accuracy for spatial differencing of Euler terms. FSO=[1,6]; non-integer values allowed. For IRHS=0, values of [2,6] are implemented: FSO=2 gives 2^{nd}-order with 4/2 dissipation; FSO=3 gives 4^{th}-order with 4/2; FSO=4 gives 4^{th}-order with 6/2; FSO=5 gives 6^{th}-order with 6/2; and FSO=6 gives 6^{th}-order with 8/2. For IRHS=2, values of [1,2] are implemented. For IRHS=3-6, values of [1,3] are implemented; FSO>3 selects WENO5 or WENO5M. [2.0 for IRHS=0,2; 3.0 for IRHS=3-6]

DELTA	MUSCL scheme flux limiter flag: For ILIMIT=1 (Koren limiter): <0.0—Turn off limiter. 0.0—Koren limiter. >0.0—Koren limiter with CFL3D-type parameter $\varepsilon=0.008\delta$. >1.0—Added smoothing with pressure/entropy switch. For ILIMIT=2-4 (minmod, van Albada, or WENO5M limiter): <0.0—Turn off limiter. 0.0-1.0—Standard limiter implementation. >1.0—Added smoothing with pressure/entropy switch. [1.0]
FILTER	0—No Q filtering. 3—3rd-order (5-point) Q filtering. 5—5th-order (7-point) Q filtering. Filtering is only done for Newton/dual time-accurate runs. [0]
EPSSGS	LU-SGS left-hand side spectral radius epsilon term (ILHS=3 only). [0.02]
VEPSL	Matrix dissipation minimum limit on linear eigenvalues. [0.0]
VEPSN	Matrix dissipation minimum limit on nonlinear eigenvalues. [0.0]
ROEAVG	Matrix dissipation flag to use Roe averaging for half-grid point flow quantities. [FALSE]

&VISINP (Viscous and turbulence modeling input)

VISC	TRUE—Include all viscous terms including cross terms. This overrides VISCJ, VISCK, VISCL and VISCX. FALSE—Include only specified or automatically enabled viscous terms. [TRUE if REY≠0, FALSE otherwise]
VISCJ	TRUE—Include viscous thin-layer terms in J. FALSE—Include viscous terms in J only if there are J-direction viscous walls. [FALSE]
VISCK	TRUE—Include viscous thin-layer terms in K. FALSE—Include viscous terms in K only if there are K-direction viscous walls. [FALSE]
VISCL	TRUE—Include viscous thin-layer terms in L. FALSE—Include viscous terms in L only if there are L-direction viscous walls. [FALSE]
VISCX	TRUE—Include viscous cross terms between coordinate directions that have thin-layer terms enabled. FALSE—No viscous cross terms. [FALSE]
WALLFUN	TRUE—Use wall function formulation for all viscous walls in this grid. FALSE—Use standard wall formulation. [FALSE]
CFLT	Turbulence model time-step is CFLT times the flow solver time-step. [1.0]
ITERT	Number of turbulence model iterations per flow solver iteration (ITER); or number of turbulence model iterations per step if ITER=0. [1]
ITLHIT	Number of subiterations for DDADI or SSOR scheme. [3 for NQT=100-102; 1 for NQT=202-203; 10 for NQT=204-205]

FSOT	1.0-1st-order differencing for turbulence convection terms. 2.0-2nd-order. 3.0-3rd-order. Intermediate values allowed; values other than 1 are only implemented for 2-equation turbulence models. [1.0 for 1-eq models; 2.0 for 2-eq models]
MUT_LIMIT	=0.0—No limit on turbulent eddy viscosity. >0.0—Maximum limit for turbulent eddy viscosity. [200,000]
IDES	0—No Detached Eddy Simulation (DES). 1—Use original DES (applies to SA or SST models). 2—Use delayed DES (DDES) (applies to SA or SST models). 3—Use delayed Multi-Scale model (D-MS) (applies to SST; SA reverts to DDES). [0]
IRC	0—No rotational/curvature correction term for turbulence model. 1—Use SARC form of rotational/curvature correction term. 2—Use approximate rotational/curvature correction term. May be applied to any 1- or 2-equation turbulence model. [0]
ICC	0—No compressibility correction. 1—Use Sarkar compressibility correction (SST model only). [1]
ITC	0—No temperature correction. 1—Use Abdol-Hamid temperature correction (2-equation models only). [0]
ITTYP(region#)	Turbulence modeling region type.
ITDIR(region#)	Turbulence model region coordinate direction (away from wall or shear layer). 1,2,3,-1,-2,-3 represent J,K,L,-J,-K,-L, resp.
JTLS(region#)	Starting J index.
JTLE(region#)	Ending J index.
KTLS(region#)	Starting K index.
KTLE(region#)	Ending K index.
LTLS(region#)	Starting L index.
LTLE(region#)	Ending L index.
TLPAR1(region#)	Turbulence model region parameter (usage depends on region type).

&BCINP (Boundary condition input)

IBTYP(region#)	Boundary condition type.
IBDIR(region#)	Boundary condition coordinate direction (away from boundary surface). 1,2,3,-1,-2,-3 represent J,K,L,-J,-K,-L, resp.
JBCS(region#)	Starting J index.
JBCE(region#)	Ending J index.
KBCS(region#)	Starting K index.
KBCE(region#)	Ending K index.
LBCS(region#)	Starting L index.
LBCE(region#)	Ending L index.
BCPAR1(region#)	Boundary condition parameter (usage depends on boundary type).
BCPAR2(region#)	Boundary condition parameter (usage depends on boundary type).
BCFILE(region#)	File name for reading boundary data (usage depends on boundary type).

&SCEINP (Species continuity input)

CFLC	Species continuity equation time-step is CFLC times the flow solver time-step. [1.0]
ITERC	Number of species continuity equation iterations per flow solver iteration (ITER); or number of species continuity equation iterations per step if ITER=0. [1]
ITLHIC	Number of species equation left-hand side subiterations: =1—Use ADI left-hand side. >1—Use SSOR left-hand side. [1]
IUPC	0—Central differencing for species convection terms. 1—Upwind differencing for species convection terms. 2—HLLC upwind differencing for species convection terms. [1]
FSOC	1.0—1st-order differencing for species continuity terms. 2.0—2nd-order. 3.0—3rd-order. Intermediate values allowed. For IUPC=0, only FSOC=2 is implemented; for IUPC=1-2, values of [1,3] are implemented. [2.0 for IUPC=0; 3.0 for IUPC=1-2]
DIS2C	2nd-order smoothing coefficient. [2.0]
DIS4C	4th-order smoothing coefficient. [0.04]

&SIXINP (6-DOF input) (OVERFLOW-D only; only for I6DOF≠2)

IBLINK	Body ID to which this grid is linked. [1]
IGMOVE	0—Body does not move (even if DYNMCS=TRUE). 1—Body motion is enabled (if DYNMCS=TRUE). [0]
IDFORM	0—Grid does not deform. 1—Grid deforms (currently only implemented for rotorcraft interface). [0]
NMAP	Update inverse-maps for deforming body every NMAP steps. [1]
NXRAY	Update X-rays for deforming body every NXRAY steps. [1]
BMASS	Body mass. [1.0]
TJJ,TKK,TLL	Body moments of inertia, about the principal axes (assumed to be body x,y,z). [1,1,1]
WEIGHT	Body weight. [0.0]
GRAVX,GRAVY, GRAVZ	Gravity unit vector (points in the direction of body weight). [0,0,1]
ISHIFT	Starting step number for applied loads (time=0). [0]
FX,FY,FZ	Body applied forces (in global x,y,z directions). [0,0,0]
FMX,FMY,FMZ	Body applied moments (about global x,y,z axes). [0,0,0]
STROKX, STROKY, STROKZ	Translation of the body CG in x, y, or z, defining the duration for applied loads to be active. [0,0,0]
STROKT	Time duration for applied loads to be active. [0.]
FREEX,FREEY, FREEZ	Enable/disable body movement in (x,y,z) directions (resp.), while applied loads are active. [TRUE]
FREER	Enable/disable (all 3) body rotational degrees-of-freedom, while applied loads are active. [TRUE]
FREE	Enable/disable all body degrees-of-freedom, while applied loads are active (sets FREEX, FREEY, FREEZ, FREER). [FALSE]

X00,Y00,Z00	Body CG location in body coordinates. [0,0,0]
X0,Y0,Z0	Initial body CG location in global coordinates. [X00,Y00,Z00]
E1,E2,E3,E4	Initial body Euler parameters in global coordinates. [0,0,0,1]
UR,VR,WR	Initial velocity of CG in global coordinates. [0,0,0]
WX,WY,WZ	Initial angular velocity about CG in global coordinates. [0,0,0]
WJ,WK,WL	Initial angular velocity about CG in body coordinates. [0,0,0]